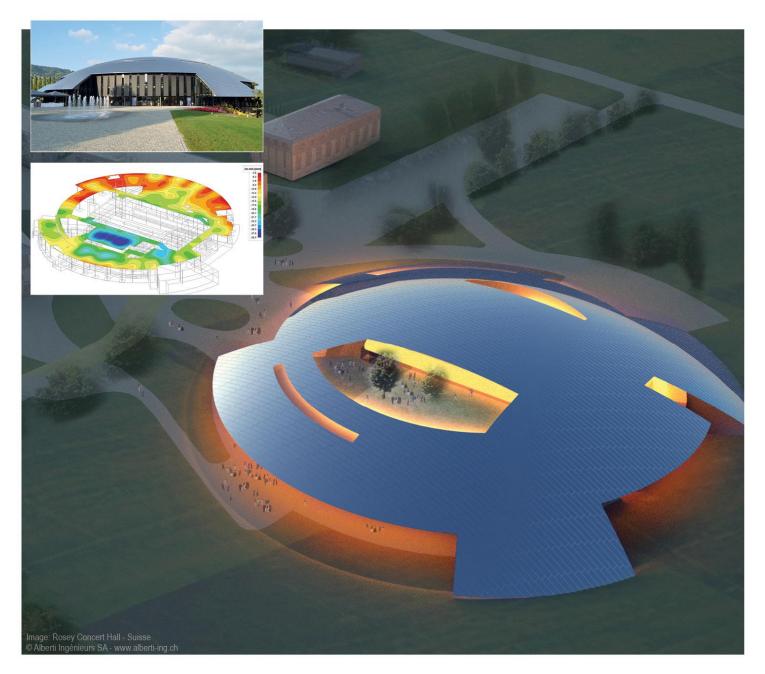
# **SCIAENGINEER**



# **Tutorial** Concrete Frame (ACI 318-08)

All information in this document is subject to modification without prior notice. No part of this manual may be reproduced, stored in a database or retrieval system or published, in any form or in any way, electronically, mechanically, by print, photo print, microfilm or any other means without prior written permission from the publisher. SCIA is not responsible for any direct or indirect damage because of imperfections in the documentation and/or the software.

© Copyright 2016 SCIA. All rights reserved.

# **Table of Contents**

General Information	5
Welcome	
Scia Engineer Support	
Website	
Introduction	
Getting started	7
Starting a project	7
Starting the program	
Starting a new project	
Project management	
Save, Save as, Close and open	
Saving a project	
Closing a project	
Opening a project	
Units	11
Changing the units	
Geometry input	
Input of the geometry	
Profiles	
Geometry	
Structure menu	13
Entering a column	14
Cursor snap settings	
Entering a beam	
Hinges	
Supports	
Check Structure data	
Checking the structure	
Connecting entities	
Graphical representation of the structure	
Loads and combinations	
Load Cases and Load Groups	
Entering a series of concentrated loads	
Entering a linear load	
Combinations	
Defining Combinations	
Calculation	
Linear Calculation	
Executing the Linear Calculation	
Results	
Viewing results	
Viewing the Reaction Forces	
Viewing internal forces on beam	
Configuring the Graphical Screen	
Code check	41
Displaying the system lengths	
Setting the Buckling Parameters	
Concrete calculation	

Displaying the Slenderness and the Buckling Lengths		45
Theoretically required reinforcement		47
Longitudinal reinforcement As	47	
Transverse reinforcement Ass	49	
Member Data	50	
Additional reinforcement	51	
Practical reinforcement	53	
Engineering Report	59	
Formatting the Report		59
Adding an image to the document	60	
Additional Engineering Report Functionality	60	

# **General Information**

# Welcome

Welcome to the Scia Engineer v15 ACI Concrete Frame Tutorial. Scia Engineer is a design program in Windows with a broad application field: from checking/designing simple frames to the advanced design of complex projects in steel, concrete, cold formed steel and a variety of other materials.

The software allows engineers to model 2D and 3D structures which include flat or curved plates and beam members (straight or curved) as well as other advanced 3D geometry. The complete calculation and design process has been integrated into one program so that the input of geometry, input of calculation information (loads, combinations, and supports), linear and non-linear calculation, output of results, reinforcement design according to various codes and the generation of the calculation documentation are all completed in the same software.

Scia Engineer is available in three different editions all which require a license to operate:

- Concept
- Professional
- Expert

#### License version

A licensed version of Scia Engineer is secured with either a 'dongle', which you apply to the USB port of your computer or a software license on your company's network.

Scia Engineer is also modular and consists of various modules. The user chooses from the available modules and composes a custom design program, perfectly tuned to the desired needs of the company.

In the general product overview of Scia Engineer you will find an overview of the different modules that are available.

#### Demo version

If the program doesn't find a license on your computer, it will automatically start the demo version. The properties of the demo version are:

- All projects can be inserted however projects created in a demo version cannot be opened in a licensed version.
- The calculation is restricted to projects with 25 elements, 3 plates/shells and two load cases
- The output contains a watermark "Unlicensed software"

# Scia Engineer Support

If you need assistance with the software, you can contact the Scia Engineer support service in the following manners:

#### By e-mail

Send an e-mail to support@scia.net with a description of the problem and the concerning \*.esa file, and mention the number of the version you are currently working with.

#### By telephone

From USA: 443-542-0637

#### Via the Scia Support website

http://www.scia-online.com/en/online-support.html

# Website

#### Link to Tutorials

http://www.Scia-online.com > Support & Downloads > Downloads > input e-mail address > Scia Engineer > Scia Engineer Manuals & Tutorials

#### Link to eLearning

http://www.scia-online.com > Support & Downloads > eLearning

#### Link to Demo version

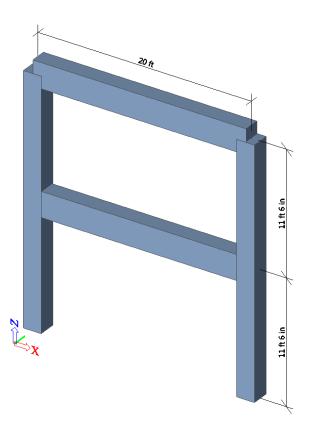
http://www.Scia-online.com > Support & Downloads > Secured Downloads > input username and password > Service Packs > Scia Engineer > Setup – Scia Engineer

# Introduction

This tutorial describes the main functions of Scia Engineer 15, the input of design data and calculation of a 2D concrete frame.

The tutorial will begin with the creation of a new project and the modelling of the concrete frame structure. After the input of all frame geometry and loads, the structure will be calculated and the results will be viewed. Next, the input of the slenderness and buckling parameters of the frame will be discussed, followed by the concrete calculations which contain the inclusion of longitudinal, transverse and additional reinforcement. Lastly, the tutorial will discuss how to format a calculation document while properly displaying the calculation results.

The figure below shows the computational model of the structure that will be completed through this tutorial (the units are in the Imperial system, e.g. kips, inch):



# **Getting started**

# Starting a project

# Starting the program

Before you can start a project, you need to start the program first.

- Double-click on the Scia Engineer shortcut in the Windows Desktop. Or:
- 2. If the shortcut is not installed, click [Start] and choose Programs > Scia Engineer 2013.1 > Scia Engineer 2013.1.

If the software cannot locate a license file, you will receive a dialogue box indicating that no license was found. A second dialogue box will then list the restrictions of the demo version. Click **[OK]** in both windows.

For this Tutorial, you must start a new project.

### Starting a new project

- 1. If the program shows the **Open** dialogue box, click [Cancel].
- 2. Click the **New** icon in the toolbar.

Select New Project							
Scia							
M						www.scia	-online.com
New project							
	<b>≜</b> ∎	$\sim$	₩ <b>I</b>				
Analysis	LTA	Scaffolder model IT	Structural Edition				
Project templates							
	code efinedShapes D Shells concrete Struc teinforced Co teel Structure (Starts	ctures ncrete	* CB-beam-ad	vanced-nen ੴ CB-be ੴ PipTa		♥ CB-slab-sta	andard-nen
Analysis - A gener drawings; it is app			delling, analysis, design	and production of out	put documents, pi	ctures and	ОК
arawings, it is app	icuble to driy	type or structure.					Cancel

In the **Select New Project** dialogue box, choose the **Analysis** environment by clicking on the corresponding icon. Confirm your choice by clicking **[OK]**.

Now, the Project data dialogue box is opened. Here, you can enter general data about the project.

	ınctionality   Loads   Code Setup   Protectio _ Data	Material	
Scia	Name:     ACI Concrete Frame       Part:     -       Description:     Tutorial - Concrete Frame       Author:     BLF       Date:     01. 10. 2012	Concrete Material C4000 Steel Timber Cther Aluminium Cther	
	Structure: Frame XZ  Project Level: Model:  Advanced  One	Code National Code:	]

- 3. In the **Basic Data** group, enter your preferred data. The data you enter will be displayed on the output produced by Scia Engineer, e.g. in the document and on the drawings.
- 4. Choose the Project level: Advanced and Model: One.
- 5. Click on the rectangular button below **National Code** to choose the default code for the project. This code will determine the available materials, combination rules and code checks. For the tutorial we will choose IBC (International Building Code). The window **Codes in project** is opened.
  - Click [Add].
  - The dialogue box Available national codes are opened.
  - Select the USA flag and click [OK].
  - You will return to the Codes in project dialogue and IBC is added.
  - Select the flag named IBC.
  - Select the Active code option and click [Close].
  - You will return to the **Project data** window and **IBC** is the active code.
- 6. Select Frame XZ in the Structure field.

The structure type (Frame XZ, Frame XYZ, Plate XY, General XYZ, etc.) will restrict the input possibilities during the calculation.

- 7. In the Material group, select Concrete. Below the item Concrete, a new item Material will appear.
- 8. Choose C4000 from the menu. This choice corresponds with the compressive design strength of the concrete.
- 9. Confirm your input with [OK].
- 10. On the **Code Setup** tab, you can see the specific codes that are used for the generation of loads and the design of steel, concrete and cold formed steel structures. In addition to these codes, you can also choose the measurement system and formatting that will be used throughout the project.
- 11. Click on **Code Setup** tab and click on the \_\_\_\_\_\_ button beside **Code setup**. Now, the **Format and measuring system** dialogue box is opened. Choose **Imperial format** and click **[OK]**.

Project data		X
Basic data Fu	unctionality   Loads   Code Setup   Protection	
Scia	Format and measuring system :	
Scia "II" Engineer	Code for loads :       ASCE 7-05         Code for concrete structures :       ACI 318(M)-08         Code for steel structures :       AISC 360-05         Code for cold formed steel structures :       AISI \$100-2007         Format and measuring system       SS         C       Metric format         @       Imperial format         Imperial system       SS         OK       Cancel	
	OK Cano	;el

#### Notes:

On the **Basic data** tab, you can set a project level. If you choose "standard", the program will only show the most frequently used basic functions. If you choose "advanced", all basic functions will be shown. On the **Functionality** tab, you choose the options you need. The non-selected functionalities will be filtered from the menus, thus

simplifying the program and the analysis.

On the **Load** tab, you will find the value for the acceleration of gravity and the applicable wind and snow loads that can be activated with the Climate loads functionality.

# **Project management**

# Save, Save as, Close and open

Before entering the software to complete the frame construction, we will first discuss how to save a project, how to open an existing project and how to close a project. While completing the tutorial, the project can be saved at any time, that way you can leave the program at any time and resume the project from the save location later.

# Saving a project

Click on in the toolbar.

If a project has not yet been saved, the dialog box **Save as** appears. Navigate to the location or the drive where you want to save your project in. Select the folder or subfolder in which you want to save the project and enter the project file name in **File name** field. Once this is complete click on **[Save]** to save the project.



If you press twice, the project is automatically stored with the same name. If you choose **File > Save as** in the main menu, you can enter a new file name or save location for the project file.

# **Closing a project**

To close a project, choose File > Close in the main menu.

A dialog box appears asking if you want to save the project. Depending on your choice, the project is saved and the active dialog box is closed.

# **Opening a project**

Click on

to open an existing project.

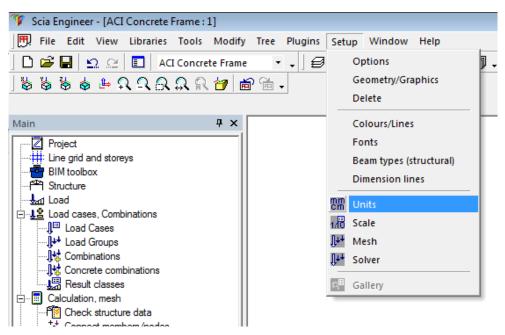
A list with Scia Engineer projects appears. Select the desired project and click [OK] (or double-click on the project to open it).

# Units

# Changing the units

At the beginning of a new project, verify that the units used by Scia Engineer are compatible with your desired units. To check and modify the units:

1. Click on Setup in the main menu and select Units.



2. In the Units window you can view the units for different options and modify them to your desired unit system.

Units		<b>-</b>	Units	<b>X</b>
Stiffness		*	□ Geometry	
Loads/Results			🗄 Length	inch
E Force	kip		🗄 Angle	deg
Unit	kip	•	Length-drawing	foot-inch
Decimal length	2		Unit	foot-inch 👻
Output format	decimal	-	Output format	fractional 👻 🧮
Stress	ksi		Fractional precision 1/2 <sup>^</sup>	3
Unit	ksi	•	Cross-section	
Decimal length	3	-	Length	inch
Output format	decimal	• E	Unit	inch 👻
🗄 Strain	1e-4		Decimal length	3
± Length-impulse	ft		Output format	decimal 👻
± Length-geom	ft		Properties	inch
Creep time	day		Unit	inch 👻
Deformation			Decimal length	3
± Mass			Output format	decimal 👻
+ Others		-	± Surface	inch^2/inch 🚽
<u>i</u> i i i i i i i i i i i i i i i i i i	OK	Cancel	<u>i e e e e</u>	OK Cancel

#### Note:

It is possible to create a default template file that contains user preferences for desired units, scales and fonts. A template file can be created and saved so that the user can load the desired template file before beginning a new project. The creation and use of template files is not discussed in this tutorial.

# **Geometry** input

# Input of the geometry

When starting a new project, the specific geometry of the structure must be entered. The structure can be entered directly, or it is possible to add geometry using instance templates with parametric blocks, DXF files, DWG files and other formats.

# **Profiles**

When entering one or more 1D structural elements, a profile type is immediately assigned to each member. By default, the active profile type is represented. At any time it is possible to open the profile library to activate another profile type. If you want to add a structural part before a profile type has been defined, the profile library will automatically be opened.

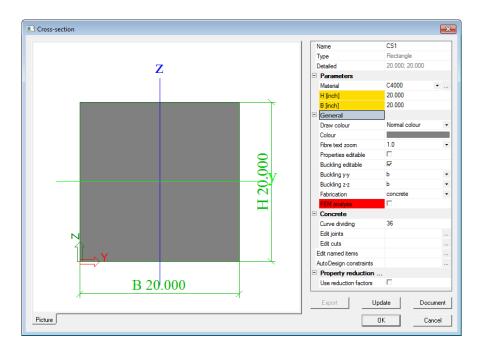
### Adding a profile

1. Click on the **Cross-Sections** icon in the toolbar.

The **Cross-Section manager** is opened. If no profiles have been entered in the project, the **New cross-section** window will be automatically opened.

New cross-section				<b>—</b> ×-
Available groups	Available items of this group			Items in project
Rectangle				
	Profile Library fi	lter	<b>_</b>	Add Close

- 2. Click Concrete in Available groups.
- 3. In the Available items of this group, you can choose a rectangular section
- 4. Click **[Add]** or  $\rightarrow$  to add the profile to the project. After adding, the **Cross-Section** window appears.

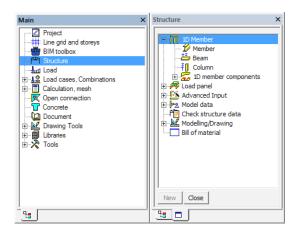


- 5. In this window, you can change the properties of the rectangular section. Enter **20 inch** for height **H** and **20 inch** for width **B**.
- 6. Click **[OK]** to confirm, the profile is added to the **Items in Project** group.
- 7. A second rectangular section with height **H** = 28 inch and width **B** = 18 inch is added in a similar way.
- 8. Click [Close] in the New Cross-Section window, the Cross-Sections Manager appears.
- 9. Click [Close] to close the Cross-Section Manager and to return to the project.

# Geometry

### Structure menu

1. When a new project is started, the **Structure menu** is automatically opened in the **Main window**. If you want to modify the structure at a later time, you must double-click on **Structure** in the **Main window** to activate the menu.

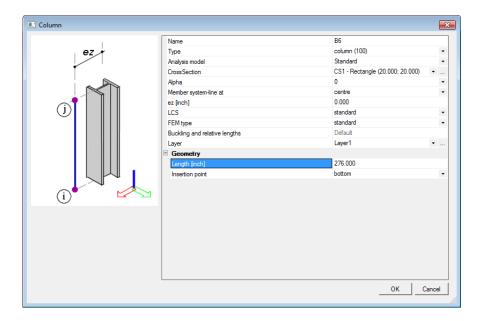


2. In the Structure menu, you can choose different structural elements to enter into the structure.

To model the frame, you first must enter the columns. After the columns have been entered, the beams on top and in the middle of the columns can be entered.

## Entering a column

1. Use the **Column** option in the **Structure Menu** to enter a new column.



- 2. In the Cross-section field, choose the concrete section CS1 Rectangle (20.000; 20.000)
- 3. Specify the column length as 276 inch or (23'-0").
- 4. The insertion position is by default set to Bottom so that the bottommost point determines the position of the column.
- 5. Confirm your input with the **[OK]** button.
- 6. The first column in the frame is positioned at the origin of the coordinate system. To accomplish this, you must enter the coordinates **0**, **0** in the **Command line** and then press **<Enter>** to confirm your input.

Com	mand lin	e			
₿	<u>s</u> s	$\vdash \overleftarrow{\rightarrow s}$	$\xrightarrow{\longrightarrow}_{X} \xrightarrow{\rightarrow}_{X}$		
New	v column	- Enter poi	int >0,0		

- 7. A second column is entered in a similar way at position 240,0 or (20', 0')
- 8. End the input using the **<Esc>** key.
- 9. After input of an entity in Scia Engineer, the entity is always selected. In this instance, the columns are colored magenta meaning they are the active selection. To cancel the selection, press the **<Esc>** key once more.

#### Notes:

The properties of selected elements are shown and can be modified in the **Properties window**. If no section has been defined in the project, the **New cross-section** window will automatically appear as soon as you try to enter a structural element (column, beam, brace, etc.). At any time, you can end your active input by pressing either the **<Esc>** key either the right mouse button.

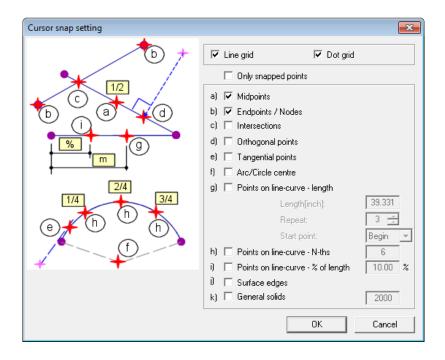
To quickly visualize the entire structure, click **Zoom All**  $\widehat{\mathbf{A}}$  in the toolbar.

When entering coordinates using the command line, separate the desired coordinates using either a comma (,) or a space.

When both columns are entered into the model, you can start adding the beams to the frame. The start and end points of the beams are already known, i.e. the center and the end point of the column. Therefore, the beams will not be entered through coordinates, but using the **Cursor Snap Settings**.

### **Cursor snap settings**

- 1. Click on the **Cursor snap settings** icon in the Command line or click on the button **Snap mode** at the lower right of the screen. The **Cursor snap settings** window is opened.
- 2. Activate the options a) and b) to select the midpoints and the end points of members as snap points in this project.
- 3. Click **[OK]** to confirm.



Now, you can enter the concrete beams.

## **Entering a beam**

- 1. To enter a new beam, you use the **Beam** option in the **Structure menu.**
- 2. In the Cross-section field, choose the second section CS2 Rectangle (28.000, 18.000)

Horizontal beam			×
	Name	B6	
	Туре	beam (80)	-
α	Analysis model	Standard	-
u l	CrossSection	CS2 - Rectangle (28.000; 18.000)	•
	Alpha	0	-
	Member system-line at	centre	-
	ez [inch]	0.000	
	LCS	standard	-
	FEM type	standard	-
	Buckling and relative lengths	Default	
🕵 ez	Layer	Layer1	•
(i)	Geometry		
$\bigcirc$	Length (inch)	240.000	
	Insertion point	begin	-
		ОК С	Cancel

- 3. Enter the beam length as 240 inch or (20'-0").
- 4. The insertion position is by default set to Begin so that the left point determines the position of the beam.
- 5. Confirm your input with **[OK]**.
- 6. Now click with the mouse on the center of the left-hand side column to enter the beam:

0				
•	Midpoint			
0				
	! ⇒x			

- 7. The upper beam is entered in a similar way by clicking the top node of the left-hand side column.
- 8. Press **<Esc>** to terminate the input.
- 9. Press **<Esc>** once more to cancel the selection.

# **Hinges**

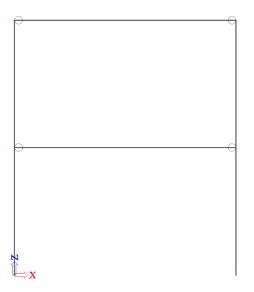
In this project, the beams are connected to the columns in a hinged manner. Since the structure type has been chosen as Frame XZ, by default the members will be connected to each other with fixed ends. Because of this, hinges must be manually added to the frame.

### **Entering hinges**

1. To enter hinges, use the **Model data > Hinge on beam** option in the **Structure menu**.

II Hinge on beam			×
	Name	H1	
	Position	Both	-
	ux	Rigid	-
	uz	Rigid	-
UX CL. ØY	fiy	Free	-
		OK Can	cel

- 2. The hinges are needed at both the beginning and the end of the beam, therefore the **Both** is selected as the **Position** of the hinge.
- 3. To obtain a hinge, the rotation fiy is taken Free, while the translations remain Fixed.
- 4. Confirm your input with **[OK]**.
- 5. The hinges are added by clicking with the left mouse button on both the top and bottom beams.
- 6. Press **<Esc>** to terminate the input.
- 7. Press **<Esc>** once more to cancel the selection.



# **Supports**

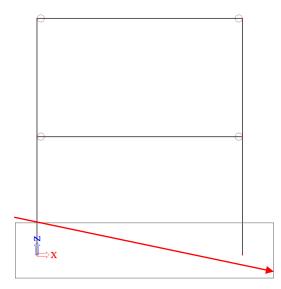
The geometry input can be completed with supports. Both column bases are modelled as fixed supports.

### **Entering supports**

1. To enter supports, use the Model data > Support > in node option in the Structure menu.

. Support in node		
	Name	Sn1
	Туре	Standard -
	Angle [deg]	
	Constraint	Fixed -
	x	Rigid -
	Z	Rigid -
×Z	Ry	Rigid -
	Default size [inch]	7.874
X	Geometry	
	System	GCS -
(i)		
$\bigcirc$		
$\hat{\mathbf{T}}$		
		OK Cancel

- 2. To model the ends as fixed, the **Constraint** is set to **Rigid**. This will set the rotation of the support in the applicable directions based on the structure type of the project (Frame XZ) to Rigid.
- 3. Confirm your input with [OK].
- 4. To specify support type on the frame, simply select both bottom nodes by drawing a box with the mouse from the left-hand side to the right-hand side:



- 5. Press **<Esc>** to terminate the input.
- 6. Press **<Esc>** once more to cancel the selection.

#### Notes:

If you draw the box from the left-hand side to the right-hand side, only entities, which are completely in the rectangle, will be selected. If you draw the rectangle from the right-hand side to the left-hand side, the entities, which are completely in the rectangle, as well as the entities that intersect with the rectangle will be selected.

The **Command line** includes a number of predefined supports. For this project, you could also have used the **Fixed Support** *icon*.

# **Check Structure data**

After input of the geometry, the input can be checked for errors by means of the option Check Structure data. With this tool, the geometry is checked for duplicate nodes, zero beams, duplicate beams, duplicate hinges and duplicate supports.

# Checking the structure

- 1. Double-click on the Check Structure data option in the Structure Menu or click on the 🛍 icon in the toolbar.
- The Structure data check window appears, listing the different available checks. 2.

Check of structure data	×
- Check of nodes	
Search duplicate nodes	Ignore parameters
Check of members	
Check members Search null members	Null members: 0  Delete null members
Search duplicate members	Duplicate members: 0 Delete duplicate members
	Invalid parts: 0 Delete invalid parts
- Check of data references	
Check data references	C Memory efficient method Fast method
Check of additional data Check additional data position	Invalid position 0
Check of steel connections	Invalid connections 0
Check load panels Check	k cross-links
Check additional data Check du	uplicity of names Check Cancel

- Click [Check] to perform the checks. 3.
- The Data Check Report window appears, indicating that no problems were found. 4.

Data check report	<b>—</b>
Data check finished. No p	problems found.
	ОК

Close the check by clicking [OK]. 5.

# **Connecting entities**

The start and end node of the top beam is an end node of a column. Therefore, this beam is automatically connected to the columns. The beam in the middle of the columns, however is not connected at a column end node. The end nodes of the beam are located at some internal point within the column and therefore are not yet connected to the columns. In this paragraph, we will explain how to connect the members to each other.

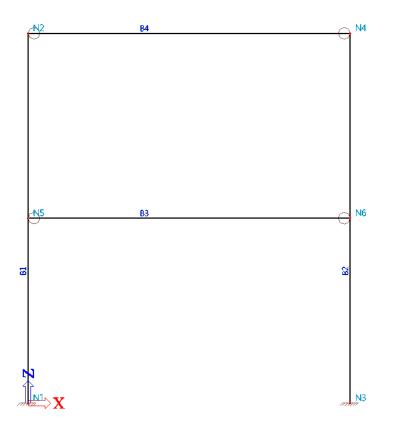
To display the names of the bars and nodes, you can activate the labels by means of the buttons in the Command line.

#### Activating node labels

Node labels are activated by means of the **Description** icon on top of the **Command line**.

### Activating member labels

Member labels are activated by means of the Ecommand line.



When you select column B1 with the left mouse button, the properties are displayed in the Property Window:

	<b></b> μ×
- Va V/	Ø
6	
B1	
column (100)	•
Standard	•
CS1 - Rectangle (20.000; 20.000)	•
0	•
centre	•
0.000	
standard	•
standard	•
Default	•
Layer1	•
276.000	
Line	
N1	
N2	
abso	
abso	
	B1 column (100) Standard CS1 - Rectangle (20.000; 20.000) 0 centre 0.000 standard standard standard tandard Layer1 276.000 Line N1 N2 abso

This window indicates that the begin node is N1 and the end node N2. Node N5 is not part of the column. To connect beam B3 to the columns, you must use the option Connect members/nodes.

### **Connecting entities**

- 1. Press <ESC> or click the Cancel selection kick icon to deactivate any selection of entities.
- 2. Double-click on the **Model data > Connect members/nodes** option in the **Structure menu** or click the icon in the toolbar.
- 3. A dialogue box appears asking if all nodes should be connected.

Scia Engineer	×
Do you want to proceed with all entities?	
Yes No Can	cel

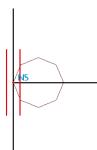
- 4. Click [Yes].
- 5. The Setup for connection of structural entities dialogue box now appears.

Setup for connection of structural entities		<b>—</b> ———————————————————————————————————
	Align structural entities to planes (moving nodes	
	Align	
	Geometrical tolerance	
	Min. distance of two nodes, node to curve [inch]	0.039
	Max. distance of node to 2D member plane [inch]	0.004
	Connect (generate linked nodes, intersections, intersections)	nt
	Connect	
	Connect 1D members as ribs	
T Restore Tel	Connect 1D members with rigid arms	
	Max. length of rigid arm [inch]	3.937
	Create new linked node for master node	V
	Check structure data	
	Check (merge duplicate nodes, erase invalid entities)	
🗟 🖻 🖬		OK Cancel

- 6. Confirm the settings by clicking [OK].
- 7. A window appears to indicate the number of connected nodes:

Scia Engin	eer	×
i	2 nodes have been successfully connected to selected members.	
	ОК	

8. Connected nodes are represented in the graphical screen by means of double red lines:



If you select beam B3, the Property Window will show that node N5 connects the beam to column B1 and that node N6 connects the beam to column B2.

Member (1)	- Va V/	1
		6
Name	B3	
Туре	beam (80)	•
Analysis model	Standard	•
CrossSection	CS2 - Rectangle (28.000; 18.000) -	
Alpha	0	•
Member system-line at	centre	•
ez [inch]	0.000	
LCS	standard	•
FEM type	standard	•
Buckling and relative I	Default 👻	
Layer	Layer1 -	
Geometry		
Length [inch]	240.000	
Shape	Line	
Beg. node	N5	
End node	NG	
Nodes		
N5	to B1	
N6	to B2	
Data		
Hinge on beam	H1	

#### Note:

If a possible active selection is not deactivated when the **Connect members/nodes** command is used, the program will only search the nodes to be connected in this selection and not in the entire project.

It is also possible to perform the two previous operations at once. To do this you have to select the option **Check (merge duplicate** nodes, erase invalid entities) in the Setup for connection of structural entities dialogue box.

9. Click [Close] at the bottom of the Structure menu to return to project view.

# **Graphical representation of the structure**

### Edit view

Within Scia Engineer there are several possibilities to edit the graphic representation of the construction. Below you will find the most important options:

- Edit the view point on the Construction
- Set a view direction
- Use the magnifier
- Edit view parameters through the menu View parameters

### Editing the view point on the construction

One of the methods for editing the view point of the structure is through the three wheels in the bottom right of the graphic window; two are horizontal and one is vertical. With these **wheels** you can **zoom in** on the construction or **turn** it.

1. To zoom in on the construction or to turn the structure, click on the appropriate wheel (the cursor will change into a hand), keep the left mouse button pressed and move the wheel

OR

Set the view point by combining the buttons and mouse.

- 2. Press CTRL + right mouse button at the same time and move the mouse to turn the construction.
- 3. Press SHIFT + right mouse button at the same time and move the mouse move the construction.
- 4. Press CTRL + SHIFT + right mouse button at the same time and move the mouse to zoom in or out on the construction.

#### Note:

If the structure is being turned while a node is selected, the structure will turn around the selected node.

### Setting a view direction with regard to the global coordinate system

- 1. Click on the button View in direction- X for a view the structure in the X-direction.
- 2. Click on the button View in direction- Y



- 3. Click on the button **View in direction- Z**
- for a view the structure in the Z-direction.

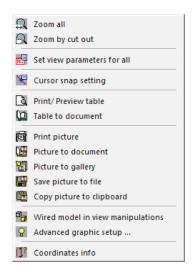
### The magnifier

- Use \Lambda to enlarge the view size.
- Use I to decrease the view size.

- Use to zoom in on a window.
  Use to view the whole structure.
- Use 😡 to zoom in on the selection.

### Editing view parameters through the menu View parameters

1. Click in the graphic window on the right mouse button. The following shortcut menu appears:



#### <u>Note:</u>

If an element was selected previously, you can define a setting that only applies to the selected elements. (An adapted shortcut menu appears).

2. Choose the option **Set view parameters for all**. The window **View parameter setting** appears. The menu consists of various tabs. You can set the view parameters for all entities or just for the selected entities.

#### **View parameters – Entities**

Through the tab entities the representation of the different entities can be adapted.

In the group Structure the following items are important for this project:

- Style and color: You can display the colors per layer, material, cross-section or structural type.
- Draw cross-section: With this the symbol of the cross-section is displayed on every member.
- Local axes: With this the local axes of the elements are activated.

View	View parameters setting		
	Check / Uncheck group Lock position		
	Modelling/Drawing   🚱 Attributes   🌌 Misc.   🔍 View		
	P Structure All Labels Model		
	Check / Uncheck all		
	Service		
	Display on opening the service	✓	
	Structure		
	Style + colour	normal 🗾	
	Draw member system line		
	Member system line style	system line 📃 💽	
	Model type	analysis model 📃	
	Display both models		
	Member surface		
	Rendering	wired 🗾	
	Draw cross-section		
	Cross-section style	section 🗾	
	Structure nodes		
	Display		
	Mark style	Dot 🗾	
	Member parameters		
	System lengths		
	Member nonlinearities		
	FEM type		
	Joists		
	Local axes		
	Nodes		
	Members 1D		
	OK	Cancel	

# View parameters – Labels and description

Through the tab **Labels**, the labels of different entities can be displayed. In the group **Members** the following items can be displayed as a label on the structure:

- Name: Show the name of the cross-sections in the label.
- Cross-section type: Show the cross-section type in the label.
- Length: show the length of the member in the label.

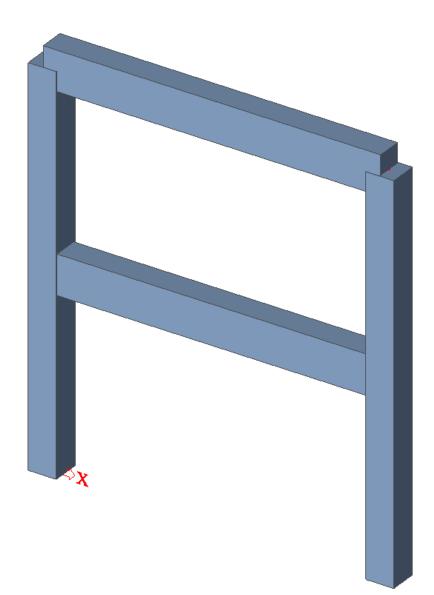
	Check / Uncheck group 2 Modelling/Drawing   🚱 Attrib	Lock position
	Structure	
_		
	Check / Uncheck all	
L	Service	
_	Display on opening the service Beam labels	
	Display label	
	Name	
	Name Cross-section name	
	Cross-section type	
	Length	
	Layer	
	Type and priority	
	Nodes labels	
	Display label	
	Name	
	X-coordinate	
	Y-coordinate	
	Z-coordinate	
	System lengths	
-	Display label	
	Name	
	Label	
F	Nonlinearities	
_	Display label	Γ
F	Labels of local axes	
_	Nodes	Γ
	Members 1D	Π
Ξ	General structural shape	
	Display vertex label	
		P

### View parameters – shortcuts

In the tool bar above the Command line, several frequently used options are grouped among which:

- **Show/hide surfaces** to show the surfaces of the cross-sections.
- **Render geometry b** to view the rendered members in the structure.
- **Show/hide supports** to show supports and hinges.
- Show/hide load **built** to show the load case.
- Show/hide other model data E to show other model data (like hinges, internal nodes, etc.)
  - Show/hide node labels 480 to view the label of the nodes.
- Show/hide member labels to view the label of members.
- Set load case for view to edit the active load case.
- Fast adjustment of view parameters on the whole construction III to quickly access the options from the View Parameters menu.

After rendering, the following structure is obtained (AXO view):



# Loads and combinations

# Load Cases and Load Groups

Each load that is inserted into the project and added to the structure is attributed to a load case. A particular load case can contain many different load types.

Each load can is attributed properties which will determine the proper generation of load combinations. In addition, a specific load case will carry a specific action which can be set as permanent or variable.

In the case of variable load cases, each variable load has its own associated load group. The group contains information about the category of the load (service load, wind, snow, etc.) and its appearance (default, together, exclusive). In an exclusive group, the different loads attributed to the group cannot act together in a normal combination. Default combinations, on the other hand, will allow for simultaneous action of the loads in the same group within the load combination generator.

The way in which load cases are defined, is critical for the load combinations created by the generator. We recommend that you thoroughly read the chapter about loads and combinations in the reference manual found in the Help menu.

In this particular project, three load cases are entered:

- LC1: Permanent Load Case: Self weight of the concrete members
- LC2: Permanent Load Case: Weight of the Floor and Weight of the Roof (Superimposed Dead Load)
- LC3: Service Load Case: Service load on the Floor (Live Load)

#### **Defining a Self Weight Load Case**

- 1. Double-click on Load in the Main window.
- 2. Before you can define loads, you first must enter load cases. Since this project does not contain any load cases yet, the Load Cases Manager will automatically appear.
- 3. By default, load case LC1 is created. This load is a permanent load of the Self Weight load type. The self weight of the structure is automatically calculated by means of this type.
- 4. In the Description field, you can describe the content of this load case. For this project, enter the description "Self Weight".

Load cases			×
🖈 💱 🥒 🛍 💺 📴	1 요   😂   😂 🖬   Ali	7	
LC1 - Self Weight	Name	LC1	
	Description	Self Weight	
	Action type	Permanent	-
	LoadGroup	LG1	•
	Load type	Self weight	-
	Direction	-Z	-
	Loads cause appreciable sidesway		
New Insert Edit	Delete		Close
	J		

#### **Defining a Permanent Load Case**

- 1. Click \_\_\_\_\_ or to create a second load case.
- 2. Enter the description "Superimposed Dead".

- 3. As this is a permanent load, change the Action type to **Permanent**.
- 4. Verify that both LC1 Self Weight and LC2 Superimposed Dead are in the LG1 LoadGroup.

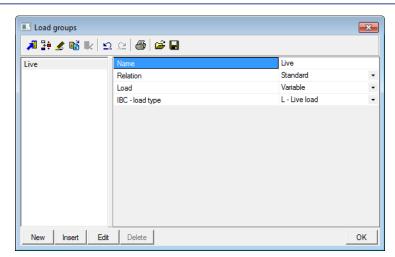
🗉 Load cases 🥃			×
A 🗄 🖉 📾 💽 🗦	🎜 🔮 🥒 📸 🔛   🕰 🖂   😂 🔚   Al		
LC1 - Self Weight	Name	LC2	
LC2 - Superimposed Dead	Description	Superimposed De	ead
	Action type	Permanent	-
	LoadGroup	LG1	·
	Load type	Standard	-
	Loads cause appreciable sidesway		
	Actions		
	Delete all loads		>>>
l	Copy all loads to another loadcase		>>>
New Insert Edit	Delete		Close

# Defining a Variable Load Case

- 1. Click New or to create a third load case.
- 2. Enter the description "Service Load".
- 3. As this is a variable load, change the Action type to **Variable**.

Load cases		<b>—</b> ×
🔎 🤮 🗶 📴 🛃	🕰 🖂   🚭   🚰 🔚   All	• 7
LC1 - Self Weight	Name	LC3
LC2 - Superimposed Dead	Description	Service Load
LC3 - Service Load	Action type	Variable -
	LoadGroup	Live 🝷
	Load type	Static -
	Specification	Standard 👻
	Duration	Short -
	Master load case	None 👻
	Loads cause appreciable sidesway	
	-	
	Actions	
	Delete all loads	>>>
	Copy all loads to another loadcase	>>>
New Insert Edit	Delete	Close

4. The LoadGroup LG2 is automatically created. Click to display the properties of the LoadGroup.



The IBC load type determines the load factors that are attributed to the load cases in this load group. In this project, choose L-live load.

- 5. Click [OK] to close the Load group manager and to return to the Load cases manager.
- 6. Click [Close] to close the Load cases manager.

#### Note:

#### Load groups

Each load is classified in a group. These groups influence the combinations that are generated as well as the code-dependant factors to be applied. The following logic is adopted throughout the software:

Variable load cases that are independent from each other are associated to different variable groups. For each group, you set the load category (see ASCE 7) and the combination factors from the American Concrete Code (ACI 318) are generated from the available load groups. When a generated combination contains two load cases belonging to different groups, reduction factors will be applied for the transient loads.

If the load is divisible, its different components are entered as individual load cases. As long as the load combination does not contain any variable load belonging to another group, no reduction factors will be applied. The different load cases of a divisible load are therefore associated to one variable group.

Load cases of the same type that may not act together, are put into one group, which is made exclusive, e.g. "Wind X" and "Wind -X" are associated to one exclusive group "Wind".

# Loads

After the input of the Load cases, the Loads menu will automatically appear:

The first load case (LC1) includes the self weight of the concrete members. As can be seen in the Loads menu, self weight is automatically accounted for and no point or line loads need to be added to the frame. The self weight is added to the structure based on the geometry and material properties of the sections.



### Switching between load cases

Activate LC2 by selecting this load case with the mouse pointer in the list box:

Load	×
LC2 - Superimposed Dead  ↓ LC1 - Self Weight LC2 - Superimposed Dead LC3 - Service Load LC3 - Superimposed Dead LC3 - Super	
New Close	

Once you have switched load cases, the floor and roof superimposed dead loads can be entered as a series of concentrated loads on the structure.

### Entering a series of concentrated loads

1. Click on Point force on beam in the Loads Menu. The dialogue box Point force on beam appears.

Point force on beam			2
$\bigcirc$	Name	F3	
	Direction	Z	•
<u> </u> ∕▼F	Туре	Force	•
	Angle [deg]		
ez	Value - F [kip]	-5.60	
	Geometry		
	Extent		٠
j⊖ <sub>x</sub> j	System		•
🖌 🖌 n x F	Coord. definition	Abso	٠
	Position x [inch]	20.000	
	Origin		٠
ez	Repeat (n)	6	
	Delta x [inch]	40.000	
	Eccentricity		
	Eccentricity ez [inch]	0.000	
(n-1) × ∆x			
		OK Cance	

- 2. The floor beam is loaded with 6 concentrated loads of **5.6 kips** with a spacing of **40 inches**. The first concentrated load in the series is at **20 inch** from begining node of the beam.
- 3. The Value of the concentrated load is changed to **-5.6 kips**.
- 4. The coordinate definition is set to Absolute.
- 5. The starting position, Position x is changed to **20 inch**.
- 6. The series consists of 6 concentrated loads, therefore set the Repeat (n) to 6.
- 7. The spacing, Delta x between the concentrated loads is **40 inch**.

- 8. Confirm your input with [OK].
- 9. Select beam B3.
- 10. Press **<Esc>** to terminate the input.
- 11. Press **<Esc>** once more to cancel the selection.

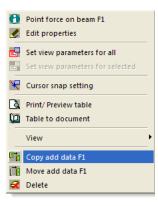
The roof beam is similarly loaded with concentrated loads of 2.8 kips. To complete this task, it is possible to copy the load from the floor beam to the roof beam and adapt it accordingly.

#### Note:

Loads, supports and hinges are considered as additional data, i.e. data that are additionally added to entities such as nodes and beams.

### **Copying loads**

- 1. Select one of the concentrated loads on the floor beam with the left mouse button. As this concentrated load is part of a series, the entire series is automatically selected.
- 2. Press the right mouse button to display a popup menu:



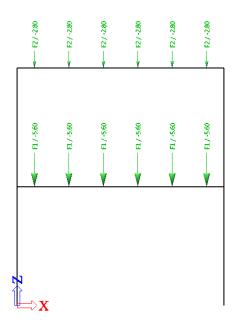
- 3. Choose the option Copy add data F1.
- 4. Select the beam where this load should be copied to: the roof beam B4.
- 5. Press **<Esc>** to terminate the input.
- 6. Press **<Esc>** once more to cancel the selection.

Use the Buick adaptation of view parameters on the entire construction icon on top of the Command line to activate the Labels of Loads option in the Loads/Masses group. A normal load is represented in green.

Now, the value of the roof load can be changed.

#### Adapting a load

- 1. Select one of the concentrated loads on the roof beam. As this concentrated load is part of a series, the entire series is automatically selected.
- 2. The properties of the series are displayed in the **Property Window**.
- 3. Change the value from -5.6 kips to -2.8 kips.
- 4. Confirm the change by pressing **<Enter>**.



After input of the loads in the LC2 load case, you can enter the service load, LC3 on the floor beam of -0.7 k/ft.

### **Entering a linear load**

- 1. Click on Line force on beam in the Loads Menu. The dialogue Line force on beam appears.
- 2. Change the type to Force and the value to -0.7 kip/ft.
- 3. Confirm your input with [OK].

	Name	LF2	
	Direction	Z	-
	Туре	Force	-
₩⇒ <b>/</b> -P2	Angle [deg]		
	Distribution	Uniform	-
	Value - P [kip/ft]	-0.70	
-P1	Geometry		
ez	System	GCS	-
	ez Location	Length	-
	Extent	full	-
	Coord. definition	Rela	-
	Position x1	0.000	
	Position x2	1.000	
	Origin	From start	-
1	Eccentricity		
L. L.	Eccentricity ez [inch]	0.000	
	- V		
		ок	Cancel

- 4. Select the beam on which this load must be positioned: floor beam B3.
- 5. Press **<Esc>** to terminate the input.
- 6. Press **<Esc>** once more to cancel the selection.
- 7. Click [Close] to leave the Loads Menu and to return to the Main Window.

#### Note:

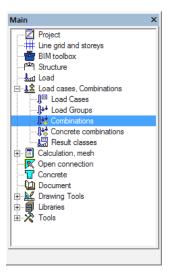
The **Command line** includes a number of predefined loads:

# **Combinations**

After input of the load cases, the latter can be grouped in combinations. In this project, two linear combinations are created, one for the Ultimate Limit State (LRFD – Ultimate) and one for the Ultimate Serviceability State (LRFD – Service).

# **Defining Combinations**

1. Double-click on 🍀 Combinations below 📲 Load cases, Combinations in the Main window.



2. Since no combination has been entered yet, the window to create a new combination will automatically appear.

Combination - Ultimate - LRFD	
Contents of combination   - Load case  - C1 - Self Weight  - LC2 - Superimposed Dead	List of load cases   Load case  LC1 - Self Weight  LC2 - Superimposed Dead
LC3 - Service Load	└─◆ LC3 - Service Load
Name : Ultimate - LRFD	Delete Add
Coeff : 1 Correct	Delete All Add All
Type : IBC (LRFD) - ultimate	
Nonlinear combination :	OK Cancel

- 3. The Type of the combination is changed to **IBC (LRFD) ultimate**. With this combination type, Scia Engineer will automatically generate combinations in accordance with the complex composition rules of the ACI.
- 4. By means of the button [Add all], all load cases can be added to the combination.
- 5. Confirm your input with [OK]. The Combination Manager is then opened.
- 6. Click New or 🚺 to create a second combination.
- 7. Change the Type of the combination to IBC (LRFD) serviceability.
- 8. Confirm your input with **[OK]**.
- 9. Click [Close] to close the Combination manager.

	🗠 🖨 Input combinations	•
Jltimate - LRFD	Name	Service - LRFD
ervice - LRFD	Description	
	Туре	IBC (LRFD) - serviceability
	Active coefficients	
	Contents of combination	
	LC1 - Self Weight [·]	1.00
	LC2 - Superimposed Dead [-]	1.00
	LC3 - Service Load [-]	1.00
	-	
	Actions	
	Actions Explode to envelopes	>>>

# Calculation

# **Linear Calculation**

If the calculation model is ready to be analyzed, it is then time to run the calculation of the frame and retrieve results.

# **Executing the Linear Calculation**

- 1. Double-click on 🛱 Calculation below 🗐 Calculation,Mesh in the Main window.
- 2. The **FE Analysis** window appears. Click **[OK]** to start the calculation.

FE analysis		<b>-X</b>
Scia	Single analysis Batch analysis	
Engineer	<ul> <li>Linear calculation</li> </ul>	
	C Nonlinear calculation	
	C Modal analysis	
	C Linear stability	
	C Concrete - Code Dependent Deflections (CDD)	
	C Construction stage analysis	
	C Nonlinear stage analysis	
	C Nonlinear stability	
	C Test of input data	
	Number of load cases: 2	
	Solver setup Mesh setup	1
	OK Cancel	
	OK Cancel	

3. After the calculation, a window announces that the calculation is finished and the maximum deformation and rotation for the normative load case is shown. Click **[OK]** to close this window.

# Results

# **Viewing results**

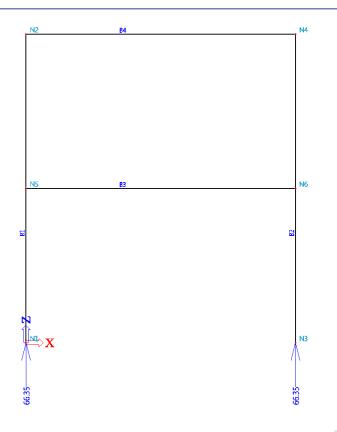
After the calculation is executed, the results of the loading on the frame can be viewed.

### **Viewing the Reaction Forces**

- Double-click on 🦊 Results in the Main window. The Results menu then appears. 1.
- Below Supports, click Reactions. 2.
- 3. The options in the **Property Window** are configured in the following way:
  - The Selection field is set to All. -
  - The Load type is set to **Combinations** and the Combination to **ULS**. \_
  - The Values are wanted for Rz. \_
  - The Extreme field is changed to Node. \_

Properties		×
Reactions (1)	- Va V	: 0
	· · · · · · · · · · · · · · · · ·	
Name	Reactions	
Selection	All	-
Type of loads	Combinations	*
Combinations	LRFD - Ultimate	Ψ.
Filter	No	•
Values	Rz	-
Extreme	Node	-
Drawing setup 1D		
Rotated supports		
Actions		
Refresh		>>>
Preview		>>>

The action **Refresh** has a red background, i.e. the graphical screen must be refreshed. Click on the behind **Refresh** to display the results in the graphical screen in accordance with the set options. 4.



5. To display these results in a table, the **Preview** action is used. Click on the **Preview** behind **Preview** to open the Preview.

#### Note:

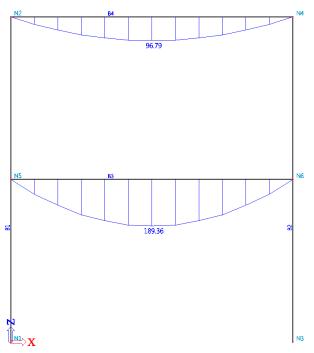
The Preview appears between the Graphical Screen and the Command line. This screen can be maximized to display more data at any time.

#### Viewing internal forces on beam

- 1. In the **Results** menu, open the **Beams** group and select **Internal forces on beams.**
- 2. The options in the **Property Window** are configured in the following way:
  - The Selection field is set to Current.
  - The Load type is set to Combinations and Combination to ULS
  - The Values are wanted for **My**.
  - The Extreme field is changed to Member.

nternal forces on mem	ber (1) 🔹 🚺	Vý 🖉
		<b>e</b>
Name	Internal forces on member	
Selection	All	-
Type of loads	Combinations	-
Combinations	Ultimate - LRFD	•
Filter	No	-
Values	My	•
Extreme	Member	-
Drawing setup 1D		
Section	All	-
Actions		
Refresh		>>>
		>>>
Detailed		

- 3. Select the two beams **B3** and **B4** using the left mouse button.
- 4. Click on the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



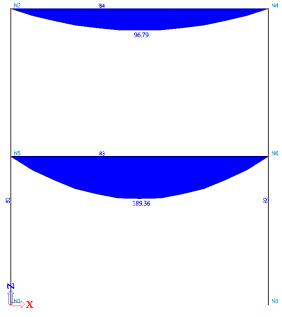
To change the display of the results, the settings of the Graphical Screen can be adapted.

## **Configuring the Graphical Screen**

1. In the **Properties window**, click the **u** icon behind **Drawing Setup**. The options for the graphical screen are opened.

Drawing setup			x
Representation : Limits :		Filled	•
B My			
Maximum [kipft]	0		
Minimum [kipft]	0		
Description			
✓ Values		🔲 Units	
🔲 Draw section in lab	els		
Draw load case or o	combination in label:	3	
Angle of text			5
C 0 deg		C User defined	
90 deg		0.00 deg	
Setup for more compon	ents		5
C Same scale		Space between diagrams	
Same height		1 ÷	
		Shift of the first diagram	
		OK Cancel	

- 2. In the Representation field, choose Filled.
- 3. The Angle of text is set to 0°.
- 4. Click **[OK]** to confirm your input.
- 5. In the **Property Window**, click the **>>>** button behind **Refresh** to display the results in the graphical screen.



- 6. Then click [Close] to leave the Results Menu.
- 7. Press <ESC> to cancel the selection.

#### Note:

To change the font size of the displayed results, you can use the **Setup > Fonts** menu. In this menu, the different sizes of the displayed labels can be changed as well as the sizes of any text associated with a specific layer.

# Code check

The concrete modules of Scia Engineer contain a number of powerful tools to execute concrete calculations in accordance with the chosen standard, in this case ACI 318-08.

The possibilities for calculations include:

- Input of advanced concrete data
- Calculation of the slenderness
- Reduction of M and V at the supports
- Design calculation of the Theoretically Required Reinforcement
- Input of the Practical Reinforcement

In this tutorial, we will only describe the basics of the concrete calculation. For more information regarding the advanced concrete calculations, please refer to the Advanced Training Curriculum.

Before you can start the concrete calculations, you first must check the buckling parameters of the beams. By means of the view parameters, the buckling lengths of the beams can be displayed on the structure.

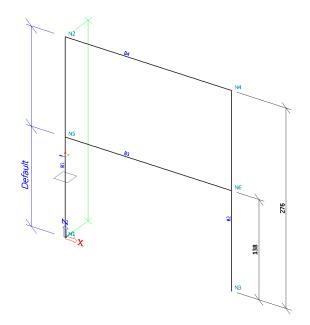
# **Buckling parameters**

## **Displaying the system lengths**

- 1. Using the left mouse button, select column B1, the left column of the frame.
- 2. Click the right mouse button at an arbitrary position in the workspace. A menu lists the possibilities for the selected entity.
- 3. In this menu, select the Set view parameters for selected option. The View parameter settings window appears.

	v parameters setting Check / Uncheck group	Lock position
	Modelling/Drawing   🚭 Attribu Structure   🔠 Labels   👗	
	Check / Uncheck all	
Ξ	Service	
	Display on opening the service	✓
	Structure	
	Style + colour	normal 🔹
	Draw member system line	✓
	Member system line style	system line 💌
	Model type	analysis model 🛛 🗸
	Display both models	
	Member surface	
	Rendering	wired 🔹
	Draw cross-section	
	Cross-section style	section -
	Structure nodes	
	Display	✓
	Mark style	Dot 🗸
	Member parameters	
	System lengths	✓
	Member nonlinearities	✓
	FEM type	<b>v</b>
	Joists	
	Mesh	
	Draw mesh	
	Free edges	
	Display mode	wired 💌
Ξ	Local axes	
	Nodes	
	Members 1D	✓
	Mesh elements	
	ПК	1

- 4. Activate the **System lengths** and **Draw cross-section** options to display the reference lengths and the cross section shape of the beam.
- 5. Activate the Local axes Members 1D option to display the local coordinate system of the beam.
- 6. Confirm your input with [OK].
- 7. Press **<ESC>** to cancel the selection.



The figure shows that system length ly for buckling around the strong axis (y-y) is 138 inch while the lz for buckling around the weak axis (z-z) is 276 inch. The beam in the middle of the column therefore supports the column for buckling around the strong axis.

To modify the buckling data of a member, use the option **Buckling and relative lengths** in the **Property window** of the selected member to change the members buckling parameters and relative length.

# **Setting the Buckling Parameters**

- 1. Select both columns with the left mouse button.
- 2. The **Properties window** shows the common properties of both entities. The **Buckling and relative lengths** are set to **Default**.

Properties	Ψ×
Member (1)	• Va V/ /
L	<b>*</b> *
Name	B1
Туре	column (100) -
Analysis model	Standard -
CrossSection	CS1 - Rectangle (20
Alpha	0 -
Member system-line at	centre -
ez [inch]	0.000
LCS	standard 👻
FEM type	standard 👻
Buckling and relative I	Default 🝷
Layer	Layer1 •
Geometry	
Length [inch]	276.000
Shape	Line
Beg. node	N1
End node	N2
Nodes	
N1	abso
N2	abso
N5	to B1
🗆 Data	
Line force on beam	LF1
Mesh data	MB1

3. Click the .... icon behind **Buckling and relative lengths**. The **Buckling data** window appears.

Buckling data			×
🗾 🎝 🤮 🗶 🖄 🔛	으 😂 🖻 🖬	Number of parts - 2	•
BC1	Name	BC1 2	
	Number of parts Member(s) material	Concrete	
		yy zz	
		•••	
		3	
		3	
1		2	
		1	
New Insert Edit	Delete		Close

This window shows that the column is supported in the middle for buckling around the strong axis (Y-Y) but not for buckling around the weak axis (Z-Z).

4. Click [Edit] to change the buckling data. The Buckling and relative lengths window appears.

Buckling and relative lengths.				×
Base settings Buckling data				
<b>X</b>	Name BC1 Buckling systems relation	Number of parts	2	
Y Y	22 = <u>22</u>	kyy	Calculate	•
		kzz	Factor	•
		Sway yy	Settings	•
Ly Lz		Sway zz	Settings	-
	Relative deformation systems relation def z = yyy v	on def y =	22 💌	
			OK Cancel	Apply

- 5. On the Base Settings tab, several data can be changed.
  - The Name field contains the name of the buckling parameter, in this case BC1.
  - **kyy** and **kzz**: in these fields, you can indicate if the program must calculate the buckling factor around the axis regarded or if you prefer entering this factor manually. A third option allows for a manual input of the buckling length. The Support option can be used to determine the buckling factor in accordance with the applicable cross section and applicable material code.
  - zz: in this field, you can indicate the system length to be used for the weak axis.
  - Sway yy and Sway zz: in these fields, you can indicate if the member is braced or not in the direction regarded. When you choose the Settings option, the default settings are used.

#### Note:

The default settings for the buckling parameters are displayed in **Concrete > 1D member > Setup > Design defaults**. By default, both directions are unbraced for a concrete calculation. So for example, in a concrete calculation, no wind bracings are assumed.

- def z and def y: in these fields, you can indicate the system length to be used for the relative deformations.

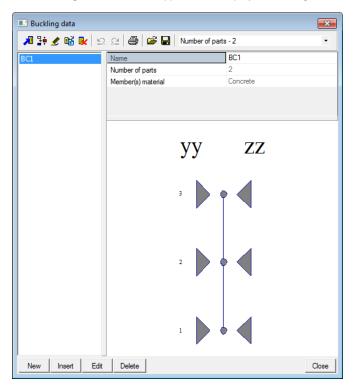
6. On the **Buckling data** tab, you can change the parameters in detail. As the columns consist of 2 components, 3 positions are available: (1) at the beginning, (2) in the center at the floor beam and (3) at the end.

For this project, we assume that the columns are not braced in both directions. Therefore, the **Non-braced** property can be set to **Yes** both for the strong and the weak axis. This could also be left on settings, since default settings are sway for concrete calculation.

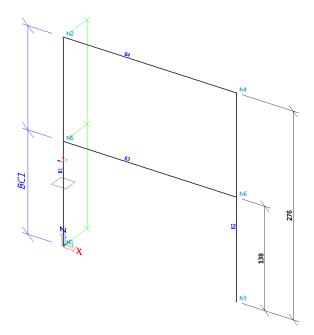
For this project, we also assume that the column is supported in the middle for buckling around the weak axis. The **zz** property at position (2) therefore can be set to **Fixed**.

	уу	Sway yy	ZZ	ł	(ZZ		Sway zz	
	Fixed	Settings	Fixed	1.00		Yes		
	Fixed	Settings	Fixed	1.00		Yes		
3	Fixed		Fixed					

- 7. Click [OK] to close this window.
- 8. The Buckling data window re-appears and displays the changed buckling data. Click [Close] to close this window.



- 9. The Properties window indicates that the buckling parameter BC1 is used for the columns in the concrete frame.
- 10. Press **<Esc>** to cancel the selection.



When the buckling parameters are set, you can continue with the concrete calculations. Before proceeding, deactivate the representation of the View parameters through Quick adaptation of view parameters on the entire construction.

When the buckling data is adapted, you must recalculate the project. For these steps, refer to the section of the tutorial concerning the Linear calculation.

# **Concrete calculation**

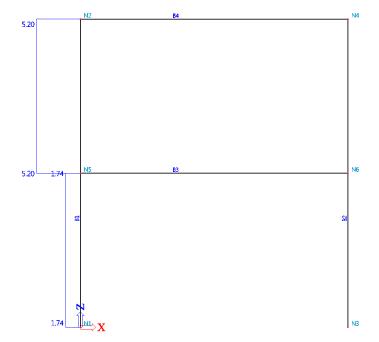
Double-click on Concrete in the Main window to open the Concrete menu.

# **Displaying the Slenderness and the Buckling Lengths**

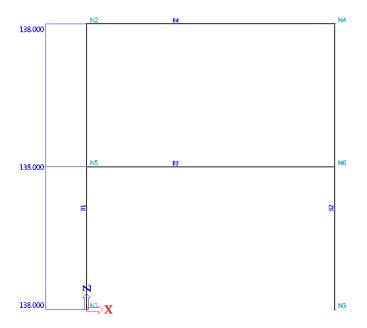
- 1. Click the Concrete slenderness icon under the **1D member tree**
- 2. The options in the **Properties window** are configured in the following way:
  - The Selection field is set to Current.
  - The Load type is set to Combinations and Combination to LRFD Ultimate.
  - The Values are desired for ky, i.e. the slenderness around the yy axis.
  - The Extreme field is modified to No.
- 3. Select column **B1**, the left column of the frame.

Properties	×
Concrete slenderness (1)	- 74 🎶 /
	<b>8</b> 🙈
Name	Concrete slenderness
Selection	Current -
Type of loads	Combinations -
Combinations	LRFD - Ultimate 👻
Filter	No 👻
Buckling coefficient de	Linear calculation -
Print explanation of err	
Values	ky 🔹
Extreme	No 👻
Drawing setup 1D	
Actions	
Refresh	>>>
Preview	>>>

4. In the **Property Window**, click the **>>>** button behind **Refresh** to display the results in the graphical screen in accordance with the set options.

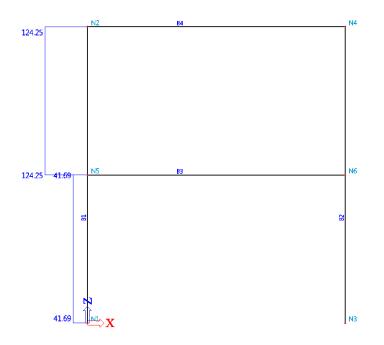


- 5. Change the Values field to luy to display the reference length for buckling around the strong axis.
- 6. In the **Property Window**, click the **>>>** button behind **Refresh** to display the new set values.



As already indicated in the buckling parameters, the reference length is 138 inches.

- 7. Change the Values field to ky\*luy/ry to display the buckling length for buckling around the strong axis.
- 8. In the **Property Window**, click the **>>>** button behind **Refresh** to display the new set value.



#### Note:

The calculation of slenderness is important for reinforcement calculation in columns. It will determine whether a second order moment has to be taken into account. For a detailed explanation of this, refer to the Advance Concrete Training tutorials.

# **Theoretically required reinforcement**

The design of theoretical reinforcement can be found under

In this tutorial the reinforcement calculation will only be completed and illustrated for the beam B3. However, it is understood that the reinforcement calculation for the other members is similar. For background on the ACI reinforcement calculations, refer to the document, Design of Reinforcement for ACI 318.

#### Note:

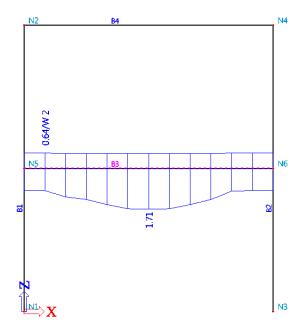
The calculation procedure for columns is different than for beams. Beams will produce an output for upper and lower reinforcement. Column reinforcement is detailing as being on one or both sides, depending on the calculation type which can be uni-axial or bi-axial bending. This theory goes beyond the scope of this tutorial, for more information, reference the Design of Reinforcement for ACI 318.

### Longitudinal reinforcement As

- 1. In the concrete menu, go to -- Hember design Design.
- 2. Press **<Esc>** to cancel the selection.
- 3. The options in the **Property Window** are configured in the following way:
  - The Selection field is set to Current.
  - The Type of load is set to Combinations and Combination to LRFD Ultimate.
  - The Values are wanted for As total req.
  - The Extreme field is changed to Member.

Design As ACI 318(M)-08	s (1) 🔹 🖓	\$7. A
Design As ACT STO(M)-00	·(I) · U	υ <del>&gt;</del> ν
		ぞ 🥓
Name	Design As ACI 318(M)-08	
Selection	Current	-
Type of loads	Combinations	-
Combinations	LRFD - Ultimate	-
Filter	No	-
Print explanation of err	M	
Values	As total req.	-
Extreme	Member	-
Drawing setup 1D		
Section	All	-
Actions		
Refresh		>>>
Calculation info		>>>
Concrete setup		>>>
New reinforcement		>>>
Single Check		>>>
Single Check Preview		>>>

- 4. Select the beam **B3** with the left mouse button.
- 5. Click the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



6. Click the button behind **Preview** to display the results on the graphical screen in accordance with the set options.

Member	d <sub>x</sub> [inch]	Case	Ф [-]	Nu [kip]	M <sub>uy</sub> [kipft]	M <sub>uz</sub> [kipft]	c [inch]	d [inch]	A <sub>s, add</sub> [inch <sup>2</sup> ]	A <sub>s,user</sub> [inch <sup>2</sup> ]	Reinf.[no.]
B3		LRFD - Ultimate/2	0.90	0.00	189.36	0.00	2.854	25.750	0.82	0.88	2x0.750+2x0.750(1.77)

7. Click the >>> button behind **Single Check** to display the calculation detailed information.

#### Note:

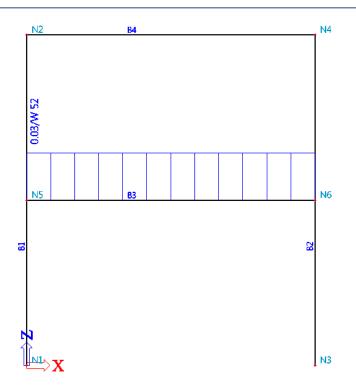
- As, req is the theoretically required reinforcement.
- Reinf.[no.] represents the reinforcement proposal by Scia Engineer (based on the default diameter set in concrete setup)
- W/E: these are warnings and errors which serve as a guideline to give extra information to the user.
  - It is possible during the calculation that also warnings and/or errors appear. If you tick on **Calculation info** in the property window, an explanation of the applicable errors or warnings appear in the preview. The aim of this errors and warnings is to give extra information to the user at this point. The most common warnings are about the distance between bars which is too big or too small according to the detailing provisions and code.

### **Transverse reinforcement Ass**

- 1. In the concrete menu, go to Hember design Design
- 2. Press **<Esc>** to cancel the selection.
- 3. The options in the **Property Window** are configured in the following way:
  - The Selection field is set to **Current**.
  - The Type of load is set to Combinations and Combination to LRFD Ultimate.
  - Tick on Print explanation of errors and warnings
  - The Values are wanted for Ass.
  - The Extreme field is changed to Section

Properties		)
Design As ACI 318(M)-08	(1) • 🕥	V/ /
		<b>e</b> 🥐
Name	Design As ACI 318(M)-08	
Selection	Current	-
Type of loads	Combinations	-
Combinations	LRFD - Ultimate	-
Filter	No	-
Print explanation of err	M	
Values	Ass	-
Extreme	Member	-
Drawing setup 1D		
Section	All	-
Actions		
Refresh		>>>
Calculation info		>>>
Concrete setup		>>>
New reinforcement		>>>
Single Check		>>>
Preview		>>>

- 4. Select the beam **B3** with the left mouse button.
- 5. Click the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



6. Click the >>>> button behind **Preview** to display the results on the graphical screen in accordance with the set options.

#### Shear reinforcement for selected members

Member	d,	Case	V <sub>u</sub>	b <sub>w</sub>	d	A,	Reinf.[no.]	W/E
	[inch]		[kip]	[inch]	[inch]	[inch <sup>2</sup> /inch]		
B4	0.000	Ultimate - LRFD/3	37.87	18.000	25.750	0.03	3x0.375-12.875	52
Explanati	ion of wa	arnings and errors						

52 The shear reinforcement was designed according to code longitudinal distance of stirrups.

7. Click the >>> button behind **Single Check** to display the calculation detailed information.

### **Member Data**

Member data can be used to change the settings for each member separately. The settings of the member data have a higher priority than the general concrete setup. For instance if you want to change the default diameter for each beam separately, member data could be used.

Note:

It should be noted that even for the calculation of theoretical reinforcement the default diameter is important. This is because it determines the lever arm d which is used for the calculation of As, required.

Member data can also be used to enter basic user reinforcement for the member.

- 1. Double-click <sup>Member data</sup> in the **Concrete Menu**.
- 2. Select floor beam B3.
- 3. The window Concrete 1D Data appears.

<b>≜</b> Z	Member	B3	
ı du	Beam type	beam	-
1	Advanced mode	N	
a tou	Minimal concrete cover		
	Input for sides		
	Situation (s)	7.7.1(c)	
	Type of concrete	Cast-in-place	-
ÿ	Corrosive environments		
	Design		
	Type of strength reduction factor Phi	Calculated	
<mark></mark> tcl	Material	W60	·
	Upper		
	Number of bars (nu)	2	
	Size number	5	
	Diameter (du) [inch]	0.625	
	Type of cover	use minimal cover	•
	Concrete cover (cu) [inch]	1.500	
	E Lower		
	Number of bars (nl)	2	
	Size number	6	
	Diameter (dl) [inch]	0.750	
	Type of cover	use minimal cover	•
	Concrete cover (cl) [inch]	1.500	
	Stirrups		
	Material	W60	• .
	Basic distance (ss) [inch]	12.000	
	Size number	3	
	Diameter (ds) [inch]	0.375	
	🖻 Shear		
	Actions		
	Load default values		>>
	Concrete Setup		>>

4. In this window, the concrete properties of the beam can be changed. For example you can modify the bar diameters, concrete coverage, reinforcement material, number of bars and stirrup size and spacing. In addition, you can define the basic user reinforcement for the beam by entering the number of longitudinal bars.

For this example, enter two #5 bars (.625" DIA) for the upper longitudinal reinforcement and two #6 bars (.75" DIA) for the lower longitudinal reinforcement. Additionally, use #3 stirrups (.375" DIA) at a longitudinal distance of 12" as shear reinforcement.

- 5. Click [OK] to confirm your input. The concrete data is added to the selected bar.
- 6. Press **<Esc>** to terminate the input.
- 7. Press **<Esc>** once more to cancel the selection.

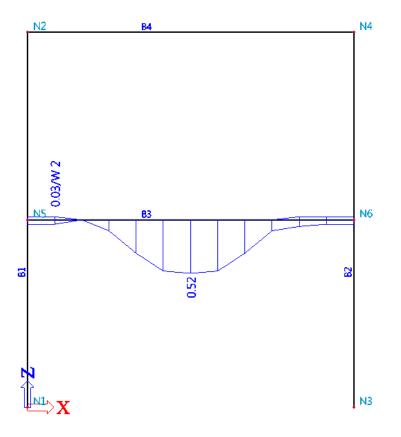
#### **Additional reinforcement**

If basic user reinforcement is input on a member, then it is possible to calculate how much reinforcement is additionally required.

- 1. In the concrete menu, go to The Member design Design
- 2. Press <Esc> to cancel the selection.
- 3. The options in the **Property Window** are configured in the following way:
  - The Selection field is set to Current.
  - The Type of load is set to Combinations and Combination to LRFD Ultimate.
  - The Values are wanted for As add.req.
  - The Extreme field is changed to Member.

Design As ACI 318(M)-08	(1) - 1/1	₩.
	00	er er
Name	Design As ACI 318(M)-08	
Selection	Current	-
Type of loads	Combinations	-
Combinations	LRFD - Ultimate	-
Filter	No	-
Print explanation of err	M	
Values	As add. req.	-
Extreme	Member	-
Drawing setup 1D		
Section	All	-
Actions		
Refresh		>>>
Calculation info		>>>
Concrete setup		>>>
New reinforcement		>>>
Single Check		>>>
Preview		>>>

- 4. Select the beam B3 with the left mouse button.
- 5. Click the >>> button behind **Refresh** to display the results on the graphical screen in accordance with the set options.



## **Practical reinforcement**

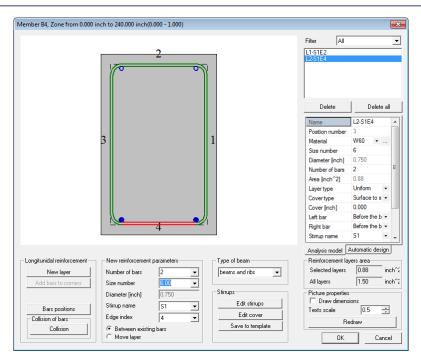
- 1. Press **<Esc>** to cancel the selection.
- 2. To enter the practical reinforcement desired, you can open the group 🔃 🖸 Redes (without As) and next select the item 💷 New reinforcement
- 3. Alternatively, you can use the shortcuts in the command line. Click the Add reinforcement on whole beam icon.
- 4. Select floor beam B3 with the left mouse button and when doing so a message box will appear to ask if the basic user reinforcement of the concrete beam should be converted into practical reinforcement.

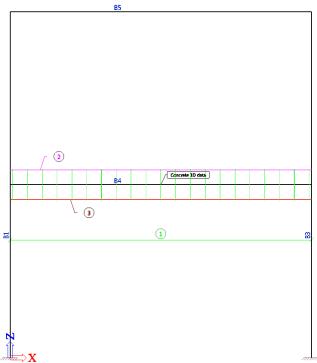
Scia Engineer		23
	r defined reinforcement? nforcement can be used only for sin	nple beam)
	Yes	No

- 5. Click **[Yes]** to accept and proceed.
- 6. The stirrup shape manager will appear. In this window, choose the default StirrupR9 and click [OK].

Stirrup shape manager	
A 19 🖌 📸 🔛 🗠 😂 🎒 🖨 🔒	
StirrupR0       StirrupR10       StirrupR12       StirrupR12       StirrupR14       Number of       Size number 3       Dameter (in 0.375       Number of       Parameters by memb *	
New Insert Edit Delete	ок

7. The next screen shows two layers of longitudinal reinforcement that are included in the cross section. The upper layer consists of 2 #5 bars (.625" DIA) and the bottom layer consists of 2 #6 bars (.75" DIA). Accept these specifications for the cross section by clicking **[OK]**.





8. In previous steps we had calculated that some additional reinforcement was required in the middle of the concrete beam. Therefore it is necessary that we add a layer of reinforcement in the zone of the beam from 3'-4" to 16'-6".

To enter an additional layer of reinforcement on a selected interval you can either use the  $\Box$  New longitudinal reinforcement icon which can be found in the concrete menu under  $\dot{\Box}$ .  $\dot{\Box}$  Redes (without As) or use the shortcut in the command line to add longitudinal reinforcement on a selected interval

To add longitudinal reinforcement on a selected interval using the shortcut, first select the  $\overline{i}$  icon and then select the beam that you want to add the reinforcement too (Beam B3 in this case). After the beam is selected you have to define the interval.

To define the interval it is helpful to use snap points on the beam. To do this, select the cursor snap settings icon  $\frac{1}{\sqrt{2}}$  in the command line. In the cursor snap settings menu activate option **h**) to create **6** snap points on each beam. Once you do this you will now have a snap point at 3'-4" increments along beam B3, now you can select an interval from 3'-4" to 16'-8" along beam B3.

9. The next screen shows us the current reinforcement layers in the zone from 3'-4" to 16'-8" along beam B3.

Member B4, Zone from 40.000 i	nch to 200.000 inch(0.167 - 0.833)	)			
	2	لم		Filter All L1-S1E2 L2-S1E4	•
				Delete	Delete all
				Name	L2-S1E4
				Position number	3
	3		1	Material	W60
				Size number	6
				Diameter [inch]	0.750
				Number of bars	2 🗉
				Area [inch^2]	0.88
				Layer type	Uniform 👻
				Cover type	Surface to s 👻
				Cover [inch]	0.000
				Left bar	Before the b 🔹
	4			Right bar	Before the b 🔹
				Stirrup name	S1
				Analysis model	Automatic design
- Longitunidal reinforcement-	New reinforcement parameters		Type of beam	- Reinforcement la	vers area
New layer	Number of bars 2	-	beams and ribs 👻	Selected layers	0.88 inch <sup>~</sup> 2
Add bars to corners	Size number 5.00			All layers	1.50 inch <sup>~</sup> 2
	Diameter [inch] 0.625		Stirrups	Picture properties	
	Stirrup name S1	_	Edit stirrups	🔲 Draw dimens	
Bars positions			Edit cover	Texts scale	0.5 📫
Collision of bars	Edge index 4	_	Save to template	Re	draw
	<ul> <li>Between existing bars</li> <li>Move layer</li> </ul>		Save to template	OK	Cancel

- 10. In this screen you can create new layers of reinforcement by using the icons in the bottom left corner of the menu. In this example input the following under New reinforcement parameters:
  - Number of bars: 1
  - Size number: 5
  - Stirrup name: S1
  - Edge index: 4

Once these new parameters are set, click on the

New layer

icon to create the new reinforcement layer.

Member B3, Zone from 40.000 ir	nch to 200.000 inch(0.167 - 0.833)			<b>—</b> ———————————————————————————————————
	3	1	Filter All L1-S1E2 L2-S1E4 L3-S1E4 Delete Name Position number Material Size number Diameter [inch] Number of bars Area [inch^2] Layer type Cover [inch] Stirup name Edge index Detailing	
Longitunidal reinforcement New layer Add bars to corners Bars positions Collision of bars Collision	New reinforcement parameters       Number of bars     I       Size number     5.00       Diameter (inch)     0.625       Stirup name     S1       Edge index     4       C     Between existing bars       G     Move layer	Type of beam beams and ribs	Analysis model	yers area 0.31 inch <sup>2</sup> 2 1.80 inch <sup>2</sup> 2

Once input, practical reinforcement layers are considered additional data. Since they are additional data, they can be easily copied to other beams. In addition, a reinforcement layers property can be easily viewed and modified by selecting the layer to open the property window.

1

- 11. We can now select the layer which contains the stirrups to modify the stirrup spacing.
  - a. Select the stirrup Reinforcement Layer by clicking with the left mouse button on the circled number 1
  - b. Selecting the stirrup Reinforcement Layer will open a Property Window showing the layers properties.
  - c. In this example we want to modify the stirrup spacing, to do this click the >>> button behind Edit Stirrup Distance.

Properties		
Stimups layer (1)	- 74	V/ /
		<b>e</b> 🔊
Name	SL	
Type of zone	stimups	
Detailing	🗆 no	
Position number	1	
Material	W60	·
Size number	3	
Diameter [inch]	0.375	
Stimups covers [inch]	1.500	
Calculation of cuts nu	From stirrup shape	-
Number of cuts	2	
Type stimup	single	-
Stimups distances [inch]	12.000	
Real distance [inch]	11.803	
Diameter of mandrel d	7	
Anchorage		
Geometry		
Description posi		
Actions		
Edit stirrup shape		>>>
Edit covers		>>>
Edit stimups distances		>>>

- d. The window for **Stirrup zones** appears.
- e. Click on the [New Part] button to enter a new section of reinforcement.
- f. Change the input type to distance + total distance.
- g. Change the Distance [in] field for this new section from 12" to 4" and chance Total Distance [in] to 36".
- h. Under the heading Minimum stirrup reinforcement, change the Distance to begin [in] to 2".

Note that the stirrups are placed closer together at both ends because the sections are automatically defined as Symmetrical.

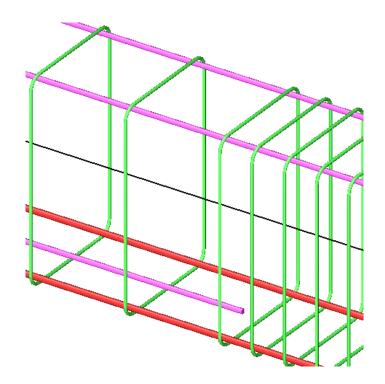
Stirrups zones												
H												++
	78.740		10-No.3(0 .000			.3(0.3 240.	2x10-No 75inch);11.' 000	· ·	omen);4	38.000	2.000	
Zone 1		creinforcement Length [inch] 240.000		Diameter (inc 0.375	h] Distance		eal distance [inch 11.714	] Type single	By user	Distance to begin [in 2.000	Text scale ch] By user Dist no	0.9 ÷ ance to end [inch] 2.000
	Additional stirru Symmetrica Parts from bo	le										-
	1	distance+t	Input type otal distance	-	Numbers 10	Size	number 3	Diameter [ir 0.375	nch]	Distance [inch] 4.000	otal distance [incl 36.000	Type single •
New zone	Delete zone	New part	Delete par	t							OK	Cancel

- i. After you are done, click [OK] to confirm your input.
- 12. The graphical representation of the practical reinforcement can also be modified so that a 3D view of the reinforcement can be obtained. To do this, complete the following steps:
  - a. Right click anywhere in the workspace.
  - b. In the menu that has opened, select the settings will appear.
  - c. On the Concrete tab, the data and view options are displayed under the Concrete + reinforcement heading. Modify the view parameters in the following way:
    - Set the option Style of stirrups to All.
    - Modify color of reinforcement to read color per diameters.
    - Set the Reinforcement drawing type to 3D.
    - Tick on the **Rounded bends** option.

0	Check / Uncheck group	Lock position			
_	🕾 Structure	AB Labels			
	🖞 Modelling/Drawing 📗 🧐 Attrib				
	👗 Model 🛛 🛃 Loads/m	asses 📅 Concrete			
$\overline{}$	Check / Uncheck all				
Ξ	Service				
	Display on opening the service				
Ξ	Concrete + reinforcement				
	Display	✓			
	Member data	~			
	SaT detail data	<b>v</b>			
	Main reinforcement	<b>v</b>			
	Style of main reinforcement	al			
	Stirrups	✓			
	Style of stirrups	al			
	Number of stirrups	al			
	Color of reinforcement	colour by diameters			
	Scheme of reinforcement				
	Reinforcement drawing type	3D			
	Rounded bends	✓			
_	Concrete labels				
Ŧ	Reinforcement labels				

d. Confirm your input by clicking [OK].

After modifying the necessary view settings, a 3D representation of the practical reinforcement is shown.



13. It is also possible to show a bill of reinforcement, which would contain a list of all the practical reinforcement on the project as well as information like the total length, weight and number of bars. To view the Bill of reinforcement, selection the icon

 Image: Bill of reinforcement
 Which can be found in the concrete menu. Once double clicking on the icon, select the Preview button in the properties window to display the following table.

#### **Bill of reinforcement**

Selection : All

The length of reinforcement and stirrups are calculated without rounded bends. Type of position number : Global

Member	Position number	Size number	Diameter [inch]	Material	Length [inch]	Number of bars	W60 length [inch]	W60 weight [kips]
B3	1	3	0.375	W60	114.377	33	3774.440	0.118
B3	2	5	0.625	W60	240.000	2	480.000	0.042
B3	3	6	0.750	W60	240.000	2	480.000	0.060
B3	4	5	0.625	W60	160.000	1	160.000	0.014
				No.3(0.375inch)			3774.440	0.118
				No.5(0.625inch)			480.000	0.042
				No.6(0.750inch)			480.000	0.060
				Total for material			4894.440	0.234
				Total			4894.440	0.234

#### Note:

Automatic member design, including reinforcement design could serve as an alternative to manual input of reinforcement

# **Engineering Report**

In this final part of the tutorial, the steps for how a calculation package can be set up and the different data sets that can be included in the Engineering Report will be discussed.

# **Formatting the Report**

1. Double-click Engineering report in the Main Window or click in the button bar. The Engineering Report appears.

The Engineering Report will open in a separate window.

2. Click the **[Insert]** button to show the New Items list. Each category can be expanded to show additional data types that can be added to the report. Each item can be added by double clicking on the item or using the **[+]** button.

New items	F 📧
計 🕂 🏠 🏦 🖽 🗮 🏨	
🗄 Scia Design Forms (standalone)	
Inbox	
Project	
🗄 · Libraries	
±. Sets	
E. Solver and Mesh	
. Structure	
tiene Load	
Construction stages	
tin Results	
🕂 Solver Files	
🕀 SGB - Scaffolding	
🕂 Special	
i ⊕ Steel	
Eustom check	
🕂 ·· Pipeline	
E Concrete	
🕀 ·· Concrete 15	
🗄 Steel concrete bridge	
. Geotechnics	
🕀 · Composite Beam	
E. Composite Column	
Here Mobile loads	
Influence lines	
🗄 Special	
Integrated Design Forms	
Gallery pictures	
E: Report templates	

- 3. By means of this window, data can be added to the report.
  - Open the Libraries group and double click on Materials.
  - Double Click Cross-Sections.
  - Open the **Structure** group and double click **Members**.
  - Open the Results group and double click Internal forces on beam.
  - Also, double click Reactions to add this item to the report.

The items that were added to the report are displayed in the **Navigator**. Drag or use the move up/down buttons to change the order of the items.

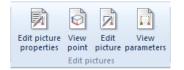
Note: For results, the properties table in the Engineering Report functions in the exact same way as it does when viewing results graphically. To see a specific result table, set the properties of the result and click the one of the regenerate buttons.

### Adding an image to the document

- 1. In the graphical interface (window) set the view of the model which you want as a picture in the Engineering Report.
- 2. Right click in the workspace and choose Screen shot into Engineering Report or Live Picture into Engineering Report.
  - Screenshot into Engineering report
    Live picture into Engineering report
- 3. The insert window will open with properties that can be set specifically for the image. Once the settings are set as desired, select one of the import options to insert the picture either directly into the report or into the report inbox.



<u>Note:</u> The main differences between a Screenshot and Live Picture are that the Live Picture is a dynamic view of the model, meaning that if the inputs or results of the model change, the Live Picture will also update automatically. Additionally, since the image is dynamic, users have the opportunity to manipulate the results of the Live Picture using the Edit Picture buttons on the ribbon.



## **Additional Engineering Report Functionality**

Modification of Tables

0

- Select specific table in Navigator and click "Edit" button. This enables the user to select and modify specific items in the table, change the table layout and also the size and spacified of the table items.
- Refresh of Engineering Report
  - After adaptations to data or (content of) tables: Red Exclamation Point means item regeneration needed.
  - Regenerate Selected Items
    - Regenerate Outdated Items (Pictures, Tables)
- Properties of the different components
  - After selection of a component in the Engineering report Navigator, its properties can be consulted and adapted in the Properties menu.

- Combination key: display of governing linear combinations
  - New items > Sets > Combination key
  - Example: Take a look at Internal forces on beam, according to Combinations = ULS; Deformations on beam, according to Combinations = SLS. In these tables with results is referred to ULS/1 etc., and SLS/2 etc. The numbers after the combination names refer to the Combination key, where the governing linear combinations are written out.
- Engineering Manager gives you the option to have multiple Engineering Reports in the same Scia file. You can also have the
  Engineering Report open in the separate window while working in the Scia file. Any changes made to the Scia file can then be updated
  by regenerating the outdated items (marked by a red exclamation point) in the Engineering Report.
- The Engineering Report also allows for any information in the "Navigator" to be saved as a report template. These templates are found in the "Report Templates" item in the List of Item.

➡·· Report templates
 ➡·· System templates
 ➡·· User templates